International Journal of Advanced Engineering Research and Science (IJAERS)
Peer-Reviewed Journal
ISSN: 2349-6495(P) | 2456-1908(O)
Vol-9, Issue-4; Apr, 2022
Journal Home Page Available: https://ijaers.com/
Article DOI: https://dx.doi.org/10.22161/ijaers.94.2

Modeling the hydrodynamic behavior of facultative pond using computational fluidodynamic tool

Leandro Conti Garcia, Flávia Aparecida Reitz Cardoso*, Flávia Vieira da Silva Medeiros, Thiago Morais de Castro and Eudes José Arantes

Post-Graduation Program of Technological Innovation (PPGIT), Federal University of Technology – Paraná (UTFPR), Campo Mourão, 87301-005, Brazil
*reitz@utfpr.edu.br

Received: 01 Mar 2022,

Received in revised form: 22 Mar 2022,

Accepted: 02 Apr 2022,

Available online: 10 Apr 2022

***Keywords*— Hydrodynamic behavior. Computational fluid dynamics. Facultative pond. Basic sanitation. Wastewater treatment.**

***Abstract*— *The stabilization pond system this which is efficient in removing organic pollutants but can present operational problems such as sediment accumulation due to lack of maintenance or operational errors, negatively influencing the efficiency of the treatment. Based on these considerations, this work evaluated the hydrodynamic behavior of an facultative pond. The hydrodynamic evaluation was carried out through the computational dynamics of the fluids, from which the main operational problems and areas of occurrence were identified. Five main regions were verified in the pond bed, with the effects of recirculation, dead zone, and short circuit zone occurring in these places. It was concluded with this study that the irregular accumulation of sediments in the pond beder and the irregular shape of the pond geometry, which contributes to the fluid is associated with the velocity of entry of the wastewater not flowing in an orderly way throughout the bed.*

I. INTRODUCTION

The most popular effluent treatment model in Brazil is composed of stabilization ponds. This system is formed by a set of ponds that have the function of removing most of the pollutants found. In this set of ponds, the facultative aerated ponds, aerated ponds of the complete mixture with decanters, anaerobic ponds with facultative or facultative ponds can be found. Each of these treatment cells has different objectives and efficiencies, which complement each other to result in an efficient treatment in the removal of physical, chemical, and biological contaminants (Chávez-Vera, 2017; Shilton; Bailey, 2006).

Von Sperling (2014) points out that the efficiency of the effluent treatment units can be compromised by the lack of maintenance in the stabilization ponds, given the high rates of sediment accumulation causing the change in the bed geometry. This situation compromises the flow of the fluid during the treatment, changing the design calculations for the hydraulic retention time, and directly influencing the pollutant removal rates.

Some of the consequences of the accumulation of sediments in ponds are the formation of dead zones, the development of short circuits, and also the expansion of thermal stratification. These phenomena can develop in parallel, and they are linked to the result found in the treated effluent (Metcalf; Eddy, 2015; Daigger, 2011).

Several studies have been carried out to solve such problems, such as the characterization of the hydrodynamic behavior of the fluid within the treatment beds and possible identification of the origin of the problem. Authors such as Passos (2017), Francener et al. (2015), Teixeira et al. (2014),Shih et al. (2017), Coggins et al. (2018),Passos, Von Sperling and Ribeiro (2014),Frederick and Lloyd (2006), and Souza et al. (2012)

use mathematical modeling applied to computational fluid dynamics to describe the movement of the fluid in the treatment ponds to find the dead zones or short circuits and other problems characteristic of sediment accumulation. Such simulation results in numerical and graphic data that corroborated the development of specific solutions and adapted to treatment systems already in operation.

According to Souza et al. (2012), during the elaboration of the projects for an effluent treatment station and dimensioning of its ponds, the ideal flow phenomena are considered. Through the characteristics pre-established in the literature, the dimensions of the project are defined. The authors also emphasize that the lack of real information on the flow of the effluent to be used does not constitute a successful operation.

Experimental data collection can often prove not to be a technically and economically feasible option, limiting the development of actions. One of the ways to obtain the necessary parameters for the correct dimensioning of the effluent treatment units is with the use of mathematical modeling (Oliveira; Teixeira, 2015).

In this respect, there is computational fluid dynamics (CFD) able to make inferences about the hydrodynamic behavior of fluids. Francener (2015) defines it as a mathematical and computational tool capable of analyzing the hydrodynamic behavior of a fluid within a predetermined volume, or outside it, in addition to relating it to aspects external to the fluid in question, such as the external environment, turbulence, pond geometry, etc.

The computational fluid dynamics function occurs through the generation of a knotted mesh established on the geometry to be analyzed. Through this mesh, equations and calculations of interest will be developed to obtain the desired result. The boundary conditions are included in the analysis platform, information referring to the type of fluid, mass movement, and inlet and outlet points, in addition to the composition material of the geometry perimeter so that the roughness and friction rates are considered (Peter, 1999).

Passos (2017) also complements the role of modeling in CFD, which: “the results obtained can be analyzed numerically and visually, (...) and compared with experimental data for calibration and validation” (Passos, 2017, p.77). The author also points out that this mathematical modeling process has advantages in relation to the others due to the reduction in the execution time of the models and projects, the cost of the study, and its high applicability in different situations involving fluid dynamic analysis.

Based on these considerations, this study proposed to solve the problem of an effluent treatment station by developing a hydrodynamic behavior model (CFD) of the facultative pond so that it was possible to analyze the points where the occur accumulation of sediments, dead zones and short circuits throughout the treatment flow.

II. MATERIAL AND METHODS

The facultative pond, object of this study, is located in the northeastern region of a city in the interior of the state of Paraná-Brazil. The treatment station where the pond is located consists of a system consisting of preliminary, primary, secondary and post-treatment. The main component responsible for the pollutants load reduction is an anaerobic reactor of the UASB type, followed by an facultative pond.

From the data obtained in the bathymetry, a three-dimensional surface was developed that characterizes the situation found at the bottom of the facultative pond. For this operation, Surfer 8® software was used. In order to be able to replicate the situation found in the pond, it was necessary to offer a treatment to the collected data, assigning values to the X, Y and Z coordinates.

Thus, the geometry of the pond was replicated in the AutoCAD® software, where it was possible to draw the collection lines and locate the points on each of the lines. After completing the design of the perimeter and collection lines, the geometry was aligned to the Cartesian axis (0.0), thus allowing the coordinates of each of the collection points to be found. Thus, the points started to have X, Y and Z dimensions.

The cartesian information generated in AutoCAD® was inserted in the Surfer spreadsheets and then made it possible to develop the lower surface of the pond. Using the tools of this software, it was possible to manipulate the generated image, grading it according to the height of the sludge layer. The three-dimensional geometry of the pond was replicated in the solid creation platform available in the Ansys 14.5® software, a high-performance program used for computer simulation.

This procedure was carried out by applying the actual dimensions of the pond for length, width, and depth, thickness, and shape of the inlet and outlet pipes. Information such as the roughness of the side and bottom walls and fluid characteristics were added to the boundary conditions provided by the *software*.

After the creation and characterization of the solid that represents the treatment pond, the Ansys® software generated a structured mesh of triangulated points with the following characteristics:

number of nodes: 369618

number of elements: 1927308

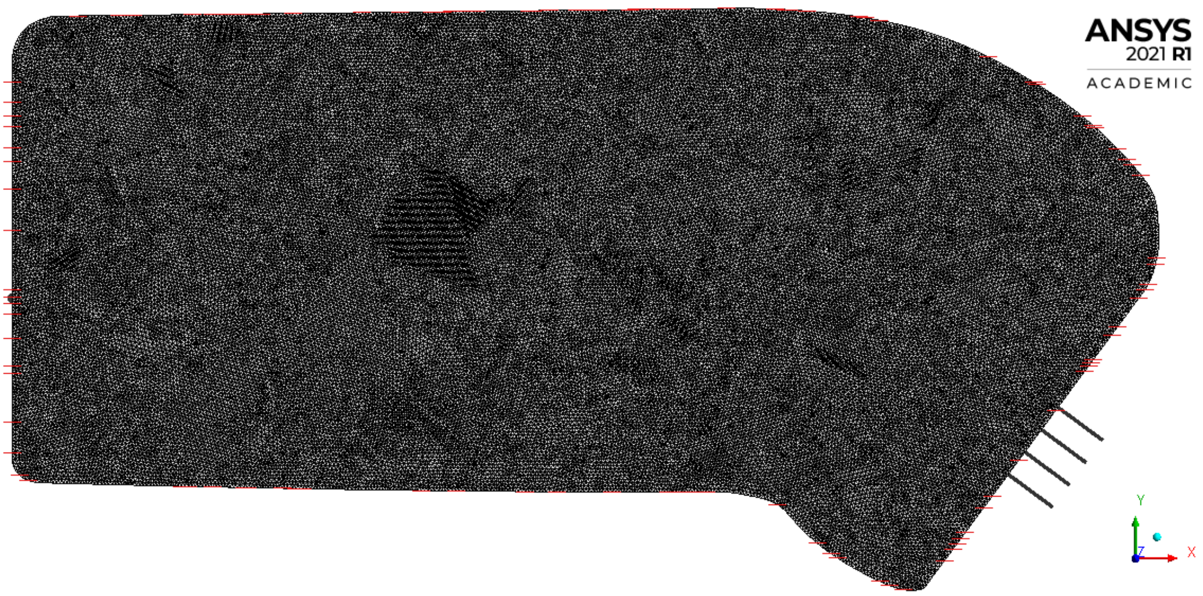
tetrahedra: 1927308

extents: min x, max x: -4.50646e-17 [m], 164.919 [m]; min y, max y: -15.4678 [m], 68.2293 [m]; min z, max z: 0 [m], 2.5 [m]

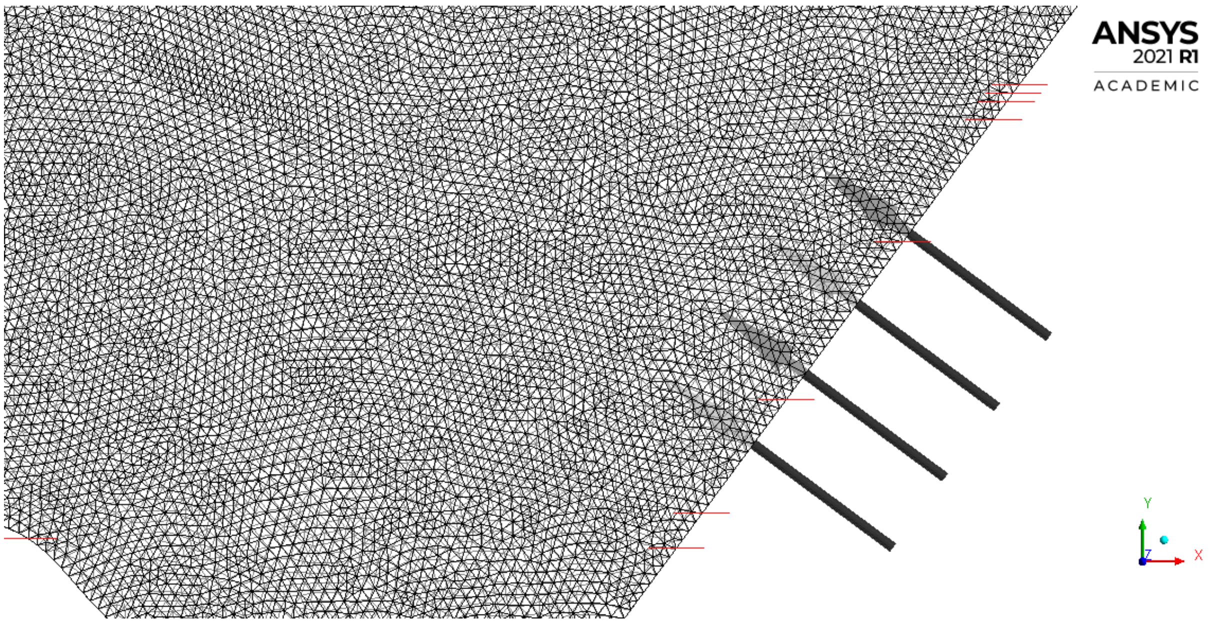
max edge length ratio: 3.59602

volume: 26913 [m^3].

To determine if the conditions of the simulation were adequate, the Courant number close to one (1) was used to prevent the fluid from passing a greater distance than the mesh size in one step of time.The details of the mesh can be better seen in Figure 1. Each node in this mesh was assigned an equation, according to the information previously provided. Also, the *software* allowed the implementation of mathematical modeling on the flow characteristics of the fluid used. The interaction of the equations in each of the points of the mesh with the fluid flow model allowed to attribute to the set (pond + effluent) the hydrodynamic behavior of the fluid along with the treatment flow, attributing to the flow a vector characterization that represents the direction, direction, and velocity of the liquid across the pond bed.



(a)



(b)

Fig.1 - Mesh details in their discretization

The results obtained in this operation were numerical (velocity reached by the fluid) and graphics (vector characterization of the fluid behavior), allowing an analysis of the existence of differences in the fluid behavior, as well as a visual analysis of the hydrodynamic vector composition. The values found refer to the fluid velocity ranges along the pond bed. This velocity range was established according to the different settings of the baffles, and already in the images, it was possible to identify the regions of short-circuit, recirculation, retrocirculation, and dead zones, which were confronted with the data collected in the field.

III. RESULTS AND DISCUSSION

From the coordinated bathymetric data, the flat (Figure 2) and three-dimensional (Figure 3) model of the sediment layer deposited at the bottom of the pond was made. Initially, a flat image was generated, representing the elevation levels found in the bed of the unit, and then a three-dimensional image of the sludge bank was created (Garcia et al., 2020).

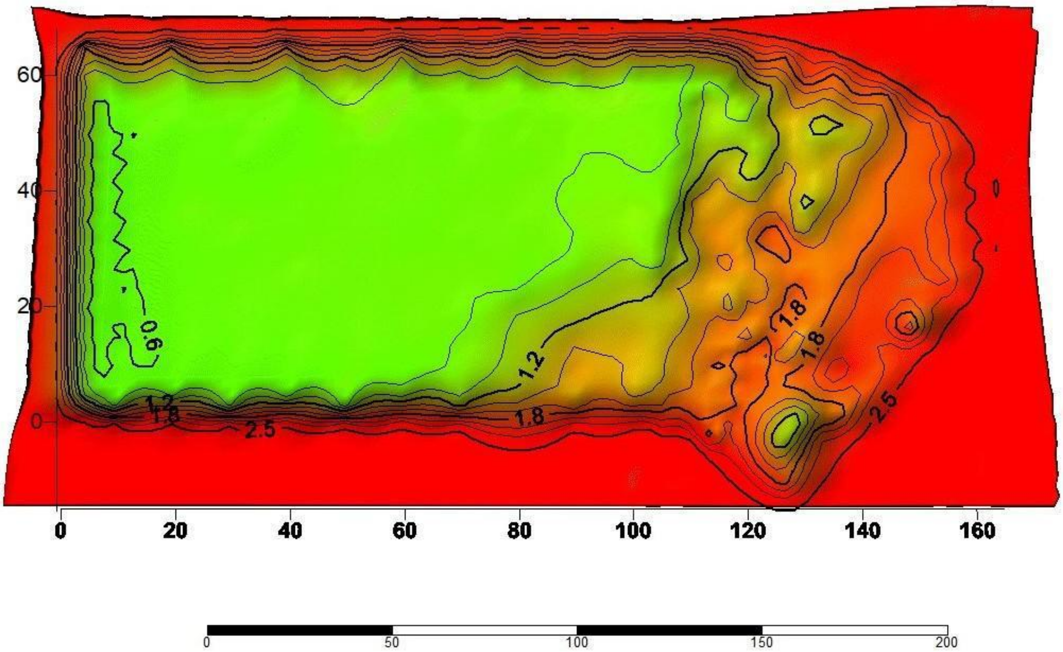


Fig.2 - Elevations of the sediment layer deposited in the bed of the facultative pond

Fonte: Garcia et al., 2020, p. 180.

Figure 2 shows that the sediment has its highest sedimentation rate at the beginning of the pond, close to the inlet ducts, reaching these points at a height of 2.5 m, the same depth as the pond (Garcia et al., 2020). The deposition pattern follows parallel to the entrance margin, in the form of a slope, where the slope gradually declines until reaching a height that varies between 0.8 and 0.6 m from the middle of the pond. It should also be noted that the sedimentation of the sludge accompanies the slope declivity, proving to be more intense close to the margins (Figure 3), as mentioned by Ortiz and Matsumoto (2013).

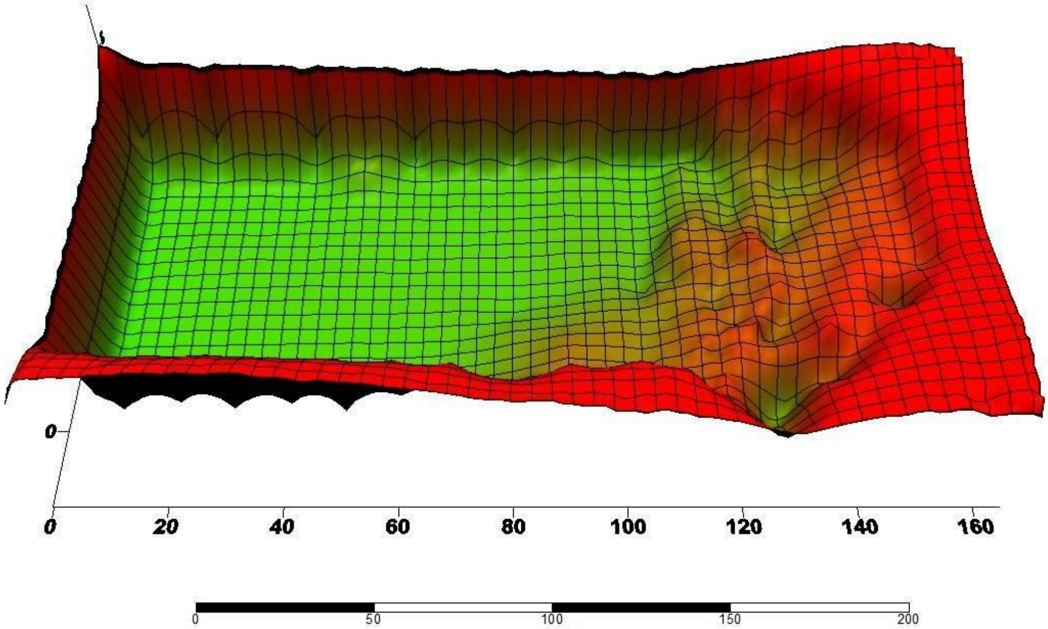


Fig.3 - Three-dimensional image and overlapping elevations of the sediment layer deposited in the bed of the facultative pond.

Fonte: Garcia et al., 2020, p. 181.

The level curves model found from the bathymetry points out that the biggest influence for the accumulation of sludge at the beginning of the pond is its irregular shape (Figure 3). This situation, based on Souza et al. (2012), is due to the fact that the curved margin becomes a barrier to the flow emitted by the inlet ducts, thus, when the liquid enters the facultative pond, it does not have a free longitudinal flow. When colliding with the curved margin, the flow rate is drastically reduced and the flow is redirected to form backmix zones. In this situation, the volume of solids thickens, conditioning sedimentation and the formation of sludge bed (Casarotti, Matsumoto; Albertin, 2012).

As in a decanter, a common primary treatment, when the fluid is able to flow through the pond, towards the outlet, the concentration of suspended solids is considerably lower, allowing the flow to occur without high sediment deposition rates. and formation of very thick sludge layers. This situation is observed in the second half of the pond, closest to the outlet ducts (Von Sperling, 2014).

Then, using computational fluid dynamics, the results obtained in this operation were numerical (velocity reached by the fluid) and graphics (vector characterization of the fluid's behavior), allowing an analysis of the existence of differences in the fluid's behavior, as well as a visual analysis of the hydrodynamic vector composition. The values found refer to the fluid velocity ranges along the pond bed and already in the images obtained, that show the behavior of the fluid at each instant of its stay inside the bed, from the inlet to the outlet, it was possible to identify the regions of short-circuit, recirculation, retrocirculation and dead zones, which were confronted with the data collected in the field.

The comparison of the surface generated by Surfer with the flow model generated by Ansys allowed to verify the reliability of the fluid dynamic modeling in the projection of the fluid current lines inside the pond, in the recirculation zones and also dead zones, places with the lowest gradients of velocity.

The velocity scale ranged from zero to 0.568 m/s. However, it remained mainly between 0.097 and 0.148 m/s. As a boundary condition of the modeling, the value of 0.137 m/s was used as the inlet velocity, calculated through the average inlet flow of the effluent, which is 28.99 L/s, thus showing that the flow velocity remains regular inside the pond, changing only when under the influence of geometry.

Figure 4 shows the intense movement of the fluid on the right margin of the pond, as well as in the space between the internal and external curves of the perimeter. These places have a lighter cyan color and increase fluid velocity. It is also observed that in the rectangular region of the pond, close to the lower margin, the vectors have a darker color, and the spacing between the posts is greater, characterizing the region as a place where the loss of flow velocity occurs (Passos, 2017).

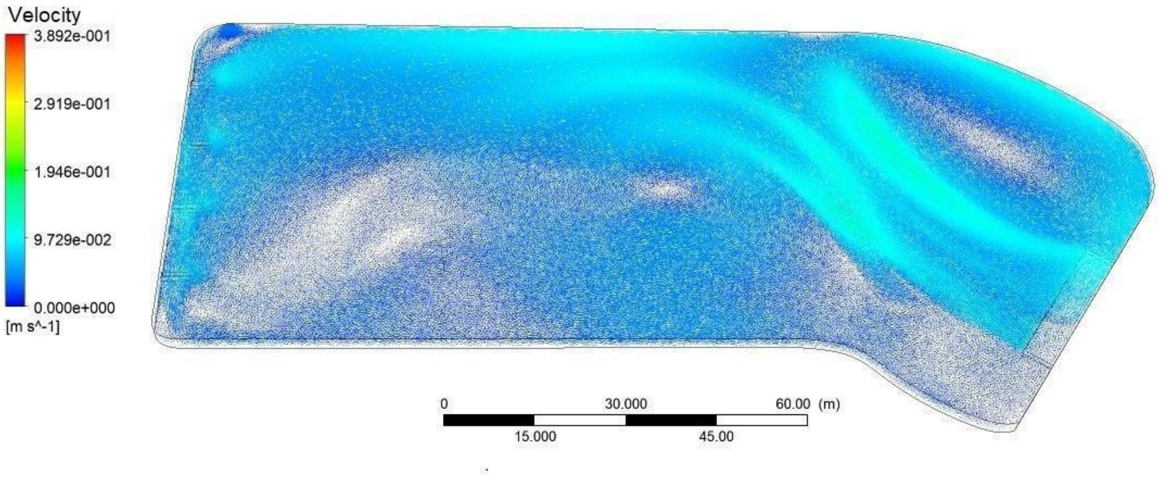


Fig.4 -Vector characterization of the flow within the facultative pond

Decomposing the vector analysis in flow lines (Figure 5), it is possible to notice with greater visibility the phenomena that affect the pond. These situations were numbered for a better explanation of each case, which can elucidate the main causes for the loss of efficiency of the treatment system in the facultative pond, as well as for the high sedimentation level near the inlet to the sewer.

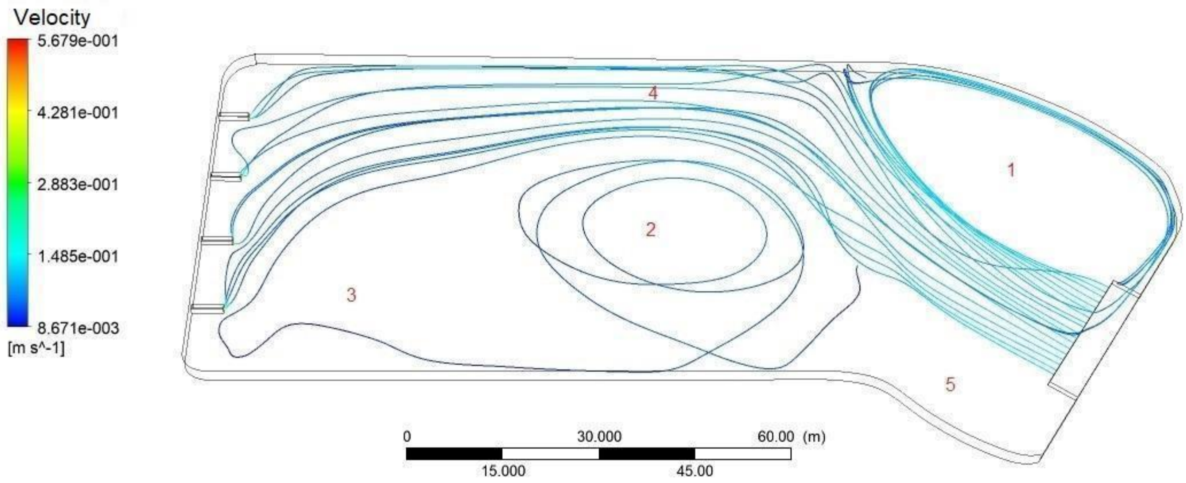


Fig.5 - Characterization of the current lines within the facultative pond

These conditions are evidence of design problems found at the site. The velocity change is caused mainly by the geometry of the location, which also determines the direction of the flow. The results demonstrate the values of velocity and stream lines, indicating the existence of dead zones and re-mixing, among others. These low velocity regions indicate hydrodynamic conditions so that the deposition occurs.

Discriminated by the number 1 (Figure 5), there is a region of retromixing, in which a portion of the liquid remains in cyclic movement until it is dragged back to the current. During this movement, there is the action of gravitational forces on the suspended solids, leading to the collision of the particles, densification of the solids, and formation of a layer of sediments. With the increase of sediments in this site, the direction of the fluid is accentuated, perpetuating this cyclical condition and therefore intensifying the adversities arising from the accumulation of sediments (Kellner; Moreira, 2009).

Another portion of the fluid, more voluminous, when finding the slope of the pond, is directed to the outlet, region number 4. In this case, the liquid follows the

barrier imposed by the pond and goes to the outlet ducts. In a pond project, the calculation for the retention time of the liquid in its interior considers that the entire volume introduced must pass through the entire bed with constant velocity to be then overflowed. When the fluid spends less time in the pond than calculated in the project, a short circuit situation is observed, where part of the fluid goes to the path with fewer obstacles to reach then the outlet (Metcalf; Eddy, 2015).

The pond analyzed has a short circuit zone (4), and even so, the operational detention time is longer than calculated. This contradiction occurs because, even though part of the fluid is directed towards the outlet more quickly than calculated, the rest of the liquid that remains in the pond has a greater volume, and this remains in it for longer than expected. One characteristic does not prevent the other from developing, as the hydraulic detention time is influenced by all other operational processes found in the pond (Jordão; Pessoa, 2014).

It is also possible to see in Figure 5 two regions of the pond where there is little or no presence of current lines. These regions received the numbers 3 and 5, and are the dead zone regions responsible for raising the hydraulic detention time above the projected. The dead zone phenomenon is characterized by a region where the fluid has a flow rate lower than the design velocity. In this case, the dead zone begins to form when the flow in the short circuit produces a centripetal force that directs the fluid to the lower margin, in a circular motion. This water flow is then guided by the slope until it finds the inlet current, which again suffers from the centripetal force produced by the short circuit. In this situation, three scenarios are obtained (Passos, 2017).

The first of them is the large area of low velocity (3), where the fluid remains almost inertial, being directed to the outlet ducts by external influence, such as the wind. The second scenario is the recirculation zone found in the middle of the pond (2). In this location, the flow that is there in a circular motion is constantly accelerated by the action of the short circuit zone, therefore having a continuous movement. The formation of the sediment layer in this region is not as intense as in the region (1), because the liquid that is in recirculation has a lower concentration of suspended solids since most of these sedimentated at the beginning of the pond (Naval, Queiro; Silva, 2002).

The third scenario is formed by a region filled with liquid, but without flow (5). At this point in the pond, there are no areas of retromixing, recirculation, or short circuit regions, and the liquid mass present there flows with almost zero velocity. In this region, can there is a large accumulation of sediments because there are characteristics similar to that of a decanter, allowing gravitational forces to act on the solid particles without the interference of the velocity gradients that would drag them to the center of the pond (Von Sperling, 2017).

The formation of this dead zone (5) is motivated by two main factors:

- The direction of the inlet flow, which conducts the fluid up to five meters into the bed of the pond and directs it in the opposite direction to that of the region.
- The recirculation region (2), which, due to its velocity greater than the fluid present in (3) and (5), does not allow a connection between these regions.

Other factors that have not been applied in the CFD simulation that may influence the gain or loss of velocity are: the concentration of sediments in the pond, the density and thickness of the sludge layer, the thermal stratification, and the intensity and direction of the winds. The factors external to the pond are capable of altering the flow close to the surface, thus influencing the way the sediments behave (Casarotti, Matsumoto; Albertin, 2012).

In cases of thermal stratification, as well as high-intensity winds, the fluid is influenced by a vertical movement, revolving the bottom sludge layer, causing the already deposited sediments to return to the suspended state, as stated by Jordão and Pessoa (2014). In the surveyed scenario, there was no evidence of the wind or thermal influence on the pond. The place where it is installed is surrounded by a curtain of large vegetation, preventing high-velocity gusts from reaching the pond, as well as protecting it from great amplitudes in the temperature variation. Historically, the city has mild temperatures, averaging around 20ºC, with positive peaks in the summer, above 35ºC, a characteristic that makes the thermal stratification effect rare (Ipardes, 2018).

The layer of sediment deposited at the bottom of the pond does not behave like a solid. The particulate material suspended in the liquid, when losing velocity, goes against the bottom of the pond, and as this process intensifies, the amount of particles increases. However, these particles remain separate from each other, so there is a space for the fluid to circulate between them. The greater the number of particles that reach the bottom of the pond, the denser the sludge, and the greater the pressure that the fluid between the particles undergoes (Kellner, Moreira; Pires, 2009).

When considering the atmospheric pressure that reaches the surface of the pond, the velocity and

turbulence of the fluid in the inlet duct, the fluid has a limited path to follow. In the analyzed pond, this path has the highest concentration of solid, the influence of these particles being the reason for the fluid velocity gain (Vos Sperling, 1996). It is important to highlight that in this specific model there are no data in the literature to corroborate the information obtained by computer simulation.

IV. CONCLUSIONS

From the analysis of the results obtained in this research, it was possible to verify the hydrodynamic behavior of the sanitary sewage in a facultative pond, in addition to the relationship between the geometry of the pond, the deposition of sediments and the development of design problems that negatively influence the efficiency rate of the pollutant removal. It was noted that the main factor influencing the deposition of solids and the formation of the sludge layer is the shape of the pond. The inlet duct directed to one of the margins conditions the collision of particles in suspension against the slope of the pond, causing a recirculation zone at the inlet of the pond and increasing the volume of sediments deposited in this region.

The collection of bathymetric data allowed the projection of the sediment layer on a three-dimensional surface, which showed the uneven concentration of solids caused by the geometry of the system. Through the bathymetry, the operational situation of the pond was actually known, and it is then possible to attest to the veracity of the mathematical model produced by the computational fluid dynamics software.

The result of computational fluid dynamics generated visual data where the main design problems can be observed, in addition to serving as a basis for decision making by the company responsible for the station for the development of mitigating activities, which enable the reduction of sediment accumulation in critical areas found here. From the development of the methodology, the main operational problems and areas of occurrence were identified, thus conditioning the choice of configurations for the future disposition of baffles.

References

[1] CASAROTTI, E.G.; MATSUMOTO, T.A.; LAZZARI, L. Avaliação da influência do ventonacirculação de umalagoa de estabilização. **EngenhariaSanitária Ambiental**, v.17, p.401-412, 2012.

[2] CELIK, I. B.; GHIA, U.; ROACHE, P. J.; FREITAS, C. J.; COLEMAN, H.; RAAD, P. E. 2008. Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. **JournalofFluids Engineering**,v. 130, n. 7, 2008.

[3] CHÁVEZ-VERA, I.G. Diseño e implementación de un sistema de tratamiento de Aguasresiduales.**Dominio de Las Ciencias**, v.3, p.536-560, 2017.

[4] COGGINS, L. X.; SOUNNESS, J.; ZHENG, L.; GHISALBERTI, M.; GHADOUANI, A. Impact of hydrodynamic reconfiguration with baffles on treatment performance in waste stabilisation ponds: full-scale experiment. **Water**, v. 10, n. 2, p. 109-130, 2018.

[5] DAIGGER, G. T. A practitioner's perspective on the uses and future developments for wastewater treatment modeling. **Water Science and Technology**,v.63, n.3, p.516-526, 2011.

[6] FRANCENER, S.F.; SOUZA, V.A.S.; PEREIRA, E.S.; BRUCHA, G.; SILVEIRA, A. Estudo da influência da hidrodinâmica no perfil de remoção de matériaorgânicaemlagoas de estabilizaçãoanaeróbia da amazôniaocidental: estação de tratamento de esgotosanitário do município de Cacoal - RO. **Engenharia Ambiental**, v.12, p.31-40, 2015.

[7] FREDERICK, G.; LLOYD, B. An evaluation of retention time and short-circuiting in waste stabilisation ponds using Serratia marcescens bacteriophage as a tracer. **Water Science and Technology**, v.33, n.7, p.49-56, 1996.

[8] GARCIA, L. C.; CARDOSO, F. A. R.; KRUGER, F. R.; GUERRA, V. S.; ARANTES, E. J. Análise do acúmulo de sedimentos por batimetriaemlagoafacultativa no município de Campo Mourão/PR. **RevistaIbero Americana de CiênciasAmbientais**, v. 11, n.1, p. 177-183, 2020.

[9] Instituto Paranaense de DesenvolvimentoEconômico e Social (Paraná). **Perfilavançado do município de Campo Mourão**. Availabe in: <http://www.ipardes.gov.br/perfil_municipal/MontaPerfil.php?codlocal=332>. Added in: 25 out. 2018.

[10] JORDÃO, E.P.; PESSOA, C.A. **Tratamento de esgotosdomésticos**. 7.ed. Abes, Rio de Janeiro, 2014.

[11] KELLNER, E.M.; VIANA, J.F.; PIRES, E.C. Influência da estratificaçãotérmica no volume útil e no comportamentohidrodinâmico de lagoa de estabilização: determinação experimental e teórica. **EngenhariaSanitária Ambiental**, v.14, p.129-138, 2009.

[12] METCALF, L.; HARRISON, E.P. **Wastewater engineering**. 5.ed. Mcgraw-Hill, New York, 2015.

[13] NAVAL, L.P.; QUEIRO, S.C.B.; SILVA, V.M. Estratificaçãotérmicaemlagoas de estabilização: influência da sazonalidade no fenômeno. In: SimpósioÍtaloBrasileiro de EngenhariaSanitária e Ambiental. **Anais...** .Abes, 2002.

[14] OLIVEIRA, D.S.; TEIXEIRA, E.C. Modelagemfluidodinâmicacomputacionalaplicadaaoprojeto de unidades de floculação. **Sodebras,** v.10, p.121-124, 2015.

[15] ORTIZ, I. S.; MATSUMOTO, T. Estudio de Batimetría y eficiencia de un sistema de lagunas de estabilización.**Revista de CiênciasAgrícolas**, v. 30, n. 1, p.65-78, 2013

[16] PASSOS, R. G.; VON SPERLING, M.; RIBEIRO, T. B. Hydrodynamic evaluation of a full-scale facultative pond by computational fluid dynamics (CFD) and field measurements. **Water Science & Technology**, v. 70, n. 3, p.569-575, 2014.
[17] PASSOS, R. G.; DIAS, D. F. C.; VON SPERLING, M. Review of practical aspects for modelling of stabilization ponds using Computational Fluid Dynamics. **Environmental Technology Reviews**, v. 5, n. 1, p.78-102, 2016.
[18] PASSOS, R.G.**Estudos e modelagem do comportamentohidrodinâmico de lagoas de polimento**.Tese. Universidade Federal de Minas Gerais, Belo Horizonte, 2017.
[19] PETER K. **Ansys**:theory reference. Canonsburg: Ansys, Inc., 1999. 1286 p.
[20] SHIH, S.; ZENG, Y.; LEE, H.; OTTE, M. L.; FANG, W. Tracer experiments and hydraulic performance improvements in a treatment pond. **Water**, v. 9, n. 2, 137-145, 2017.
[21] SHILTON, A.; BAILEY, D. Drogue tracking by image processing for the study of laboratory scale pond hydraulics. **Flow Measurement and Instrumentation**, v. 17, n. 1, p. 69-74. 2006.
[22] SOUZA, V.A.S.; HELBEL, A.F.; JANZEN, J.G.; DIAS, R.H.S.; COLTRE, H.C.S. Simulaçãocomputacional do comportamentohidrodinâmicoemlagoasfacultativas com chicanas. **Ciência&Engenharia**, v.21, p.15-21, 2012.
[23] TEIXEIRA, J.C.; OLIVEIRA, G.S.; VIALI, A.M.; MUNIZ, S.S. Estudo do impacto das deficiências de saneamentobásicosobre a saúdepública no Brasil no período de 2001 a 2009. **EngenhariaSanitária Ambiental**, v.19, p.87-96, 2014.
[24] VON SPERLING, M. **Lagoas de estabilização**.3.ed. UFMG,Belo Horizonte, 2017.
[25] VON, S.M. **Introdução à qualidade das águas e aotratamento de esgotos**.4.ed. UFMG, Belo Horizonte, 2014.